Improving Fan Impact Simulation for Debris Ingested by a Turbofan Engine

Thesis

Presented in Partial Fulfillment of the Requirements for Undergraduate Research Distinction

in Aerospace Engineering

By

Ian Chamberlain

The Ohio State University

2018

Thesis Committee:

Dr. Kiran D'Souza, Advisor

Dr. Michael Dunn

Abstract

Unmanned Aerial Vehicles (UAVs) pose a new threat to the safety of airplanes. Turbofan engines have been designed and tested to survive foreign object ingestions for decades. For example, turbofan engines are tested for their ability to withstand a bird strike. However, UAVs have components which are harder and denser than birds. These differences make it impossible to predict the damage from UAV ingestion based on a bird of the same mass. To assess the risks that UAVs may pose in an engine ingestion scenario, researchers are using finite element software, such as LS-DYNA, to simulate collisions resulting from the ingestion of a UAV into a jet engine. The goal of this project is to develop and test new meshes for use in continued research studying the damage suffered by the fans of jet engines after UAV ingestion. These new meshes aim to improve on old meshes by removing all tetrahedron elements and taking contacts out of the blades. The new meshes are then impact tested in LS-DYNA to examine their improvements relative to the previously used meshes. Impact testing is carried out using a steel bolt. The results obtained through these impact tests confirm the hypothesis that a stress discontinuity occurs across the blade contact used in the old mesh of the fan. It is also discovered that a plastic strain discontinuity also occurs across the blade contact under high energy impacts. The new mesh is successful in resolving both of these problems and is composed of about 98% hexahedron elements.

Acknowledgements

I would like to thank Dr. Kiran D'Souza for acting as my advisor for this research. This project has been a great learning experience, and I am grateful for being given the opportunity to contribute to the GTL's ongoing research in UAV ingestion.

Special thanks to Troy Lyons, who provided guidance and support throughout this project. Without the countless hours that Troy has spent teaching himself LS-DYNA, this project would not have been possible.

I would also like to thank Dr. Mike Dunn for sitting on the committee for this project and all the other members of the GTL who have helped during the course of this research.

The funding for the GTL's ongoing research into UAV ingestion is provided by the Federal Aviation Administration. Computational testing for this project was made possible in part by the Ohio Supercomputer Center.

Lastly, I would like to thank my family and all the great friends I have here at Ohio State for supporting me throughout this project and my college career as a whole. Without you, the achievements I've made in college would have been impossible.

Table of Contents

List of Figures

List of Tables

1. Introduction

Over the past several years, the use of unmanned aerial vehicles (UAVs) for commercial and personal use has expanded significantly. This increased popularity has resulted in a new class of threats for commercial and general aviation aircraft. For decades, research into midair collision events between aircraft and foreign objects has focused on bird strikes, with jet engines being required to meet specific impact survivability guidelines [1]. However, UAVs contain harder and denser materials than those found in birds which makes adapting these guidelines for UAV ingestion unfeasible.

Figure 1: Fan Damaged by Bird Ingestion [2]

As a result, research is being conducted into the effects of UAV ingestion in The Ohio State University's Gas Turbine Laboratory (GTL). As with previous work in bird ingestion, the focus of this research is on the damage caused to fan blades of a turbofan engine. The engine being examined in this case has a 40 in diameter titanium fan that would typically be seen on a regional or business jet. This work is being conducted in LSTC LS-DYNA, a commercially available multiphysics simulation software that is often used for impact simulations. This ongoing research initiative has proven that UAVs can cause significant damage to the fan, in some cases causing partial loss of multiple blades [3]. The first round of results obtained in this research has prompted the Federal Aviation Administration (FAA) to continue this work in the hopes of better understanding the danger that UAVs pose to air travel.

1.1 Motivation

The goal of this project is to continue improving the existing simulation capabilities of the GTL in support of the ongoing research into the effects of UAV ingestion. By improving these capabilities, more accurate simulation results will be obtained. These can then be used by the FAA to understand the true danger that these collisions present and help their efforts focused on preventing these collisions through a variety of technologies (e.g., detect and avoid) and regulations (e.g., restrictions on the airspace where UAVs can operate).

1.2 Scope of Work

The primary focus of this work is the development of new meshes and meshing protocols for the fan assembly being used for impact simulations. Previous meshes for this project were created using ANSYS Meshing but all meshes moving forward will be made using Hypermesh. Hypermesh is preferred because it allows for greater control over meshing parameters, making it possible to create the high quality meshes necessary for this type of simulation. New meshes are then used for trial impact simulations in LS-DYNA and compared with the previous fan meshes to see what improvements are achieved.

Initial consideration is also given to internal improvements to the MAT-224 model, which is used in these impact simulations. The MAT-224 model requires element size regularization curves, inclusive of all element sizes in the mesh being used, to function at its highest degree of accuracy. Due to the high computational cost associated with running impact simulations of this size, it is desired to use element sizes outside the current regularization curve of the MAT-224 Ti-6Al-4V model to keep element counts as low as possible. As a result, extending the regularization curve is required to obtain the best possible results. This aspect of the ongoing UAV ingestion project is still in its early stages, so all information regarding its progress thus far will be located in the Future Work section of the report.

2. Methodology

This project's goal is to improve the accuracy and efficiency of existing simulation procedures being employed to study the effects of UAV ingestion. In this section, the methods employed to improve these simulation procedures will be examined. The goal of improving the existing simulation procedure is met by:

1. Creating an improved mesh of the fan assembly
2. Comparing impact test results for new and old meshes

2.1 Mesh Creation

The main improvement made in this project is the development of a new mesh for the fan assembly being tested in LS-DYNA. The assembly consists of the fan blades, disk, and nose cone (or spinner). Two separate models for the fan blades are meshed, one representing a generic thick fan blade design (0.25 in thick at the tip) and the other representing a generic thin fan blade design (0.15 in thick at the tip). Specific improvements to be made include removing any tetrahedron element sections present in the previous mesh and removing contacts from the fan blades.

Using hexahedron elements is preferred to tetrahedrons because tetrahedron elements have been proven to demonstrate an overly stiff response under loading [4]. It is also recommended to use "significantly more [than 2] elements" in the thickness when simulating fracture behavior. [5]. These guidelines were followed when creating the updated mesh of the fan assembly, with "significantly more" being interpreted as 5 elements in thickness for the thick blades, 4 elements for the thin blades, and 4 elements for the nose cone. It is also decided that the elements at the end of the fan blades should remain at or below 1 mm in length. These element counts through the thickness satisfy the requirement of having significantly more than two elements in thickness and keep element sizes at the blade tips below 1 mm.

Finished meshes are given checks for element quality focusing of Jacobian, aspect ratio, and warpage. These parameters are checked against recommended values of >0.5, <5, and <30° respectively [6]. Particular attention is paid to the element quality in the section of the blade being impacted by the UAV.

2.2 Comparison of Impact Results

Impact testing is conducted on the thick fan model using an elastic bolt as the impact object. The elastic bolt is selected as the impactor because it reduces the computational cost significantly compared to a UAV model and imparts a large amount of damage, which is beneficial for comparing the meshes. The bolt being used has a 0.5 in diameter with a 1.25 in shaft. Simulations are conducted in LS-DYNA at two rotational velocities, 2000 rpm and 8500 rpm, with the bolt entering the fan at 180 knots. These conditions represent the engine's state during takeoff or landing, which is when ingestion of a bolt or UAV is most likely to occur.

The impact runs for both meshes are set up in LS-DYNA as follows. The fan is constrained to rotate about its axis using a nodal rigid body constraint. Contacts holding the fan together are set using a tied nodes to surface contact. An initial rotational velocity corresponding to the RPM of that specific test is applied to the fan. An initial velocity of 180 knots in the direction of the fan's axis is applied to the bolt. Eroding surface to surface contacts are set between the bolt and the blades of the fan. An eroding single surface contact is set for the fan blades to account for the possibility of them impacting themselves. The MAT-224 Ti-6Al-4V model being used for the fan in ongoing UAV ingestion testing is applied to the fan. An elastic MAT-001 model for a generic hardened steel is applied to the bolt (ρ=8 g/cc, E=200 GPa, PR=0.25). Appropriate controls are applied to the system [5].

The goal of these tests is to see how the new mesh compares to the old mesh when under impact conditions. Specific attention is paid to the interaction of the contact used in the blade of the old mesh as compared with the new mesh, which is made without any contacts in the blade. It is hypothesised that this contact in the blade will cause a stress discontinuity under loading from the bolt impact. Overall damage caused by the bolt impact, including qualitative observation of the damage and examination of the effective plastic strain and Von-Mises stress around the impact point, is also a key result that is extracted from these tests.

3. Results

3.1 Mesh Creation

The mesh creation phase of this project seeks to address two items: remedying known issues with the old mesh and meeting the meshing guidelines state in §2.1.

3.1.1 Improving on Old Mesh

Mesh creation is completed in Hypermesh. The first meshes to be examined will be those of the thick and thin blisk. Most elements used in meshing the blisks are hexahedron elements with a small number of pentahedron elements being required to keep reasonable quality throughout the mesh. The element cross sections at the blade tip for the thick and thin blades are shown in Figures 2 and 3.

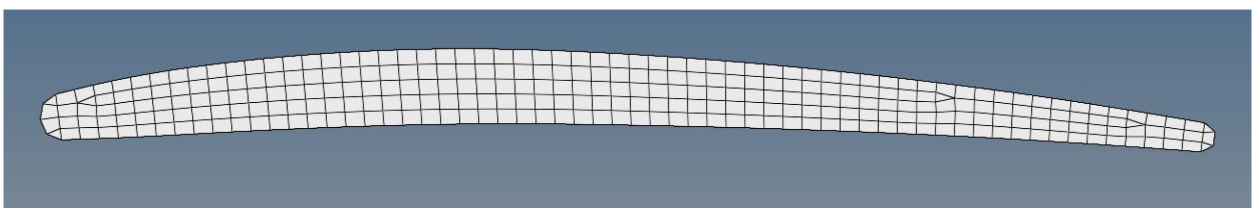

Figure 2: Thick Blade Cross Section

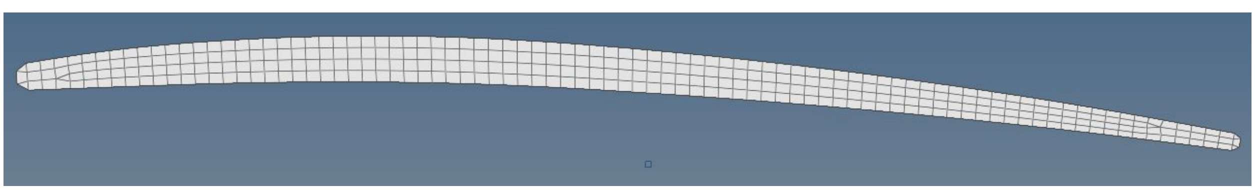

Figure 3: Thin Blade Cross Section

These meshes are created such that all the elements at the tip are approximately 1 mm in all directions. The outer 60% of each blade is also created with 1 mm elements to keep the element size reasonably small in the impact area. The elements then grow progressively larger until they reach the fan disk.

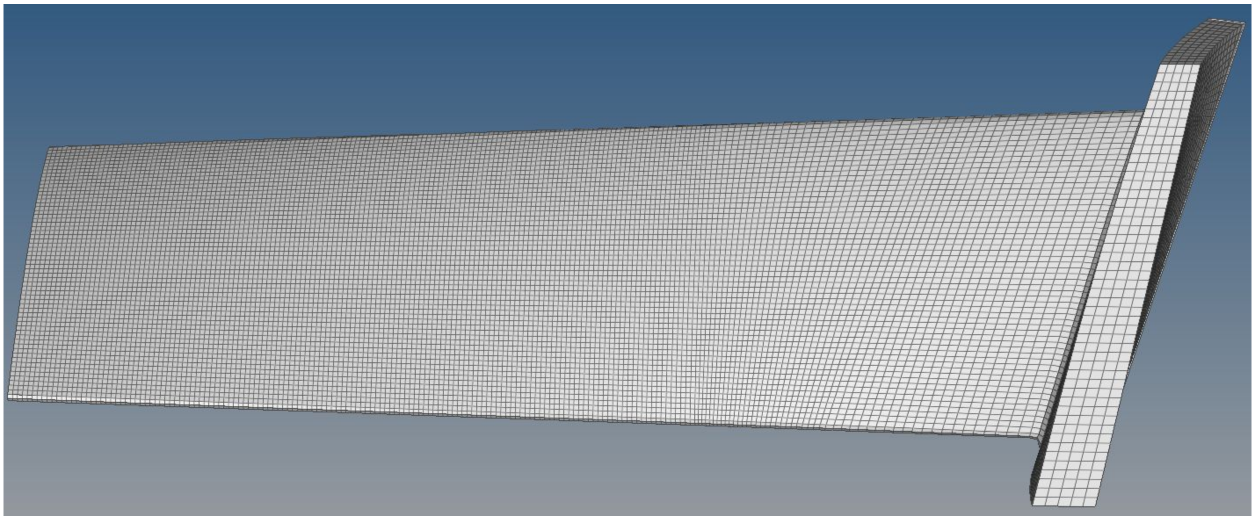

Figure 4: Thick Blade Profile

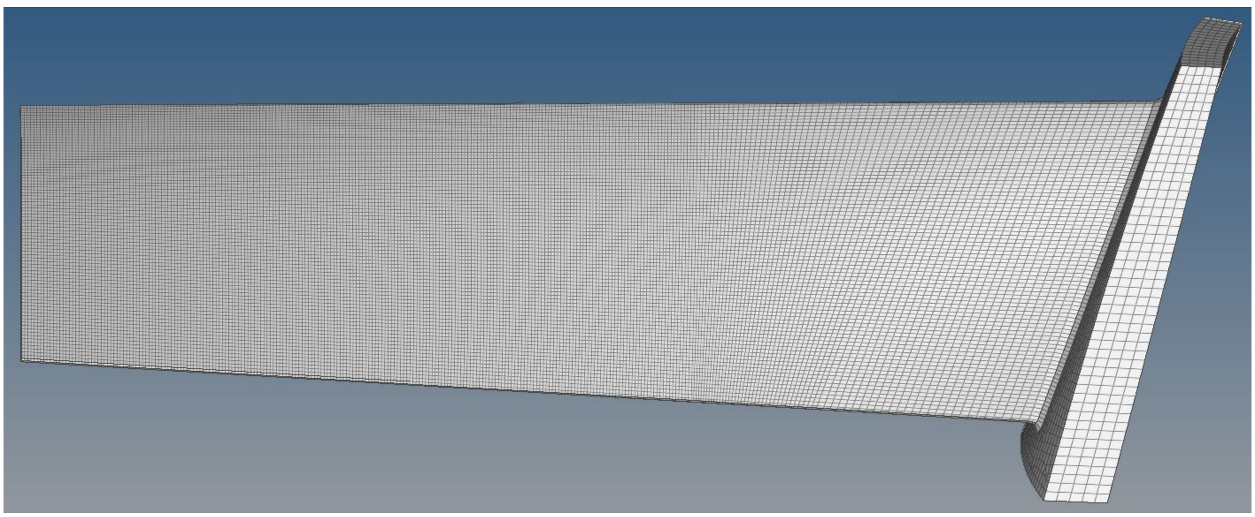

Figure 5: Thin Blade Profile

The disk itself is split into two parts; each blade shown above is copied 20 times to create the outer disk and blade part, and the inner disk part is separated. The new meshes make substantial improvement to the previous mesh both in the element cross section and by removing contacts from the blades. The old mesh is shown in Figures 6 and 7 as a reference.

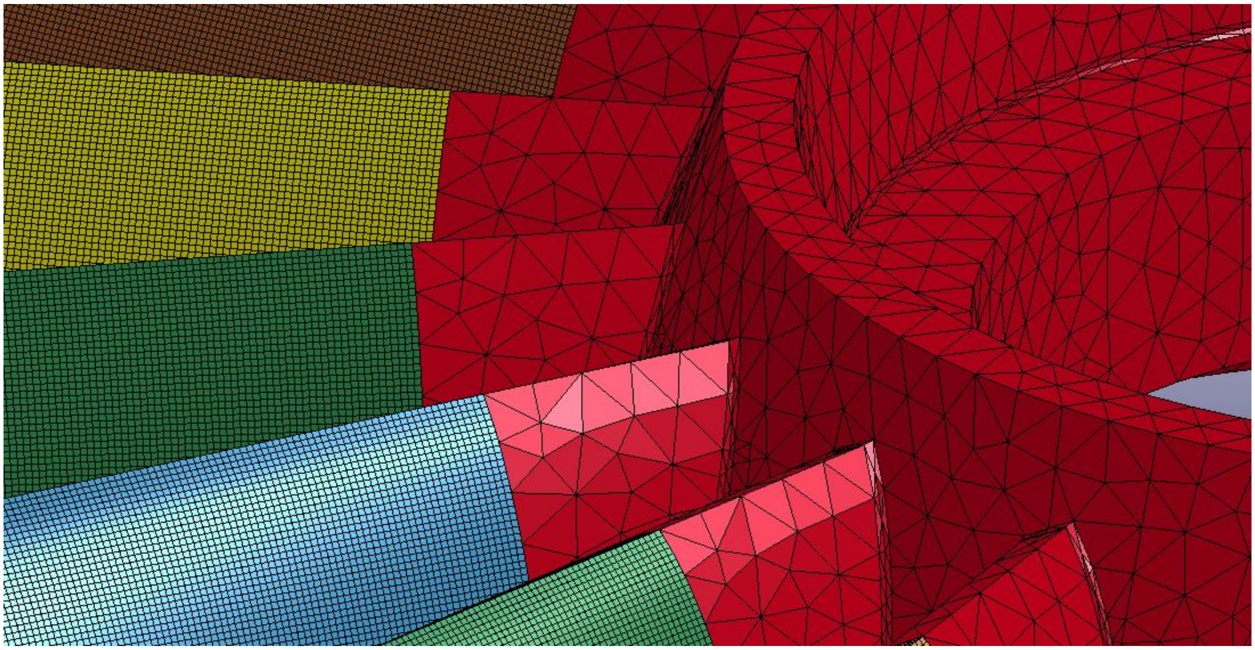

Figure 6: Contacts in Old Mesh

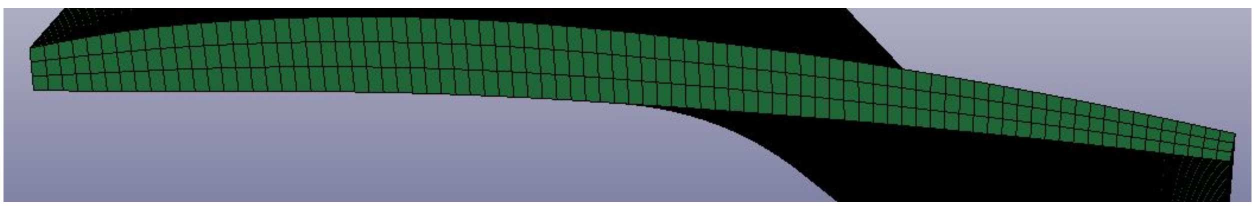

Figure 7: Cross Section of Old Mesh

Comparing the new and old cross sections, it is immediately clear that the new mesh overall has better element aspect ratios, that is the aspect ratios are closer to one. The MAT-224 model being used for material properties in the impact simulation is known to be sensitive to aspect ratio [7], so making this improvement should provide better results when impact testing is conducted. A comparison of the aspect ratios for each mesh is given in Figures 8 and 9. The contours show how the new mesh manages to keep the aspect ratio closer to the ideal of one throughout most of the fan.

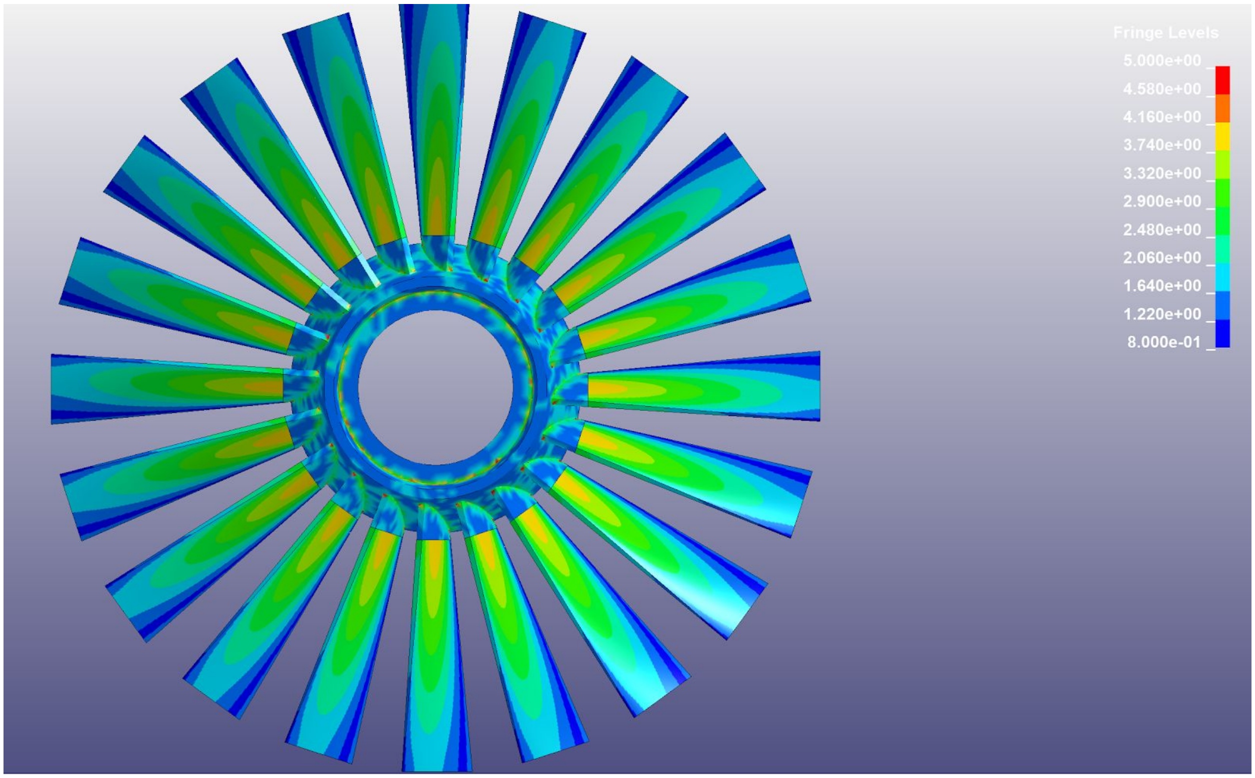

Figure 8: Aspect Ratio in Old Mesh

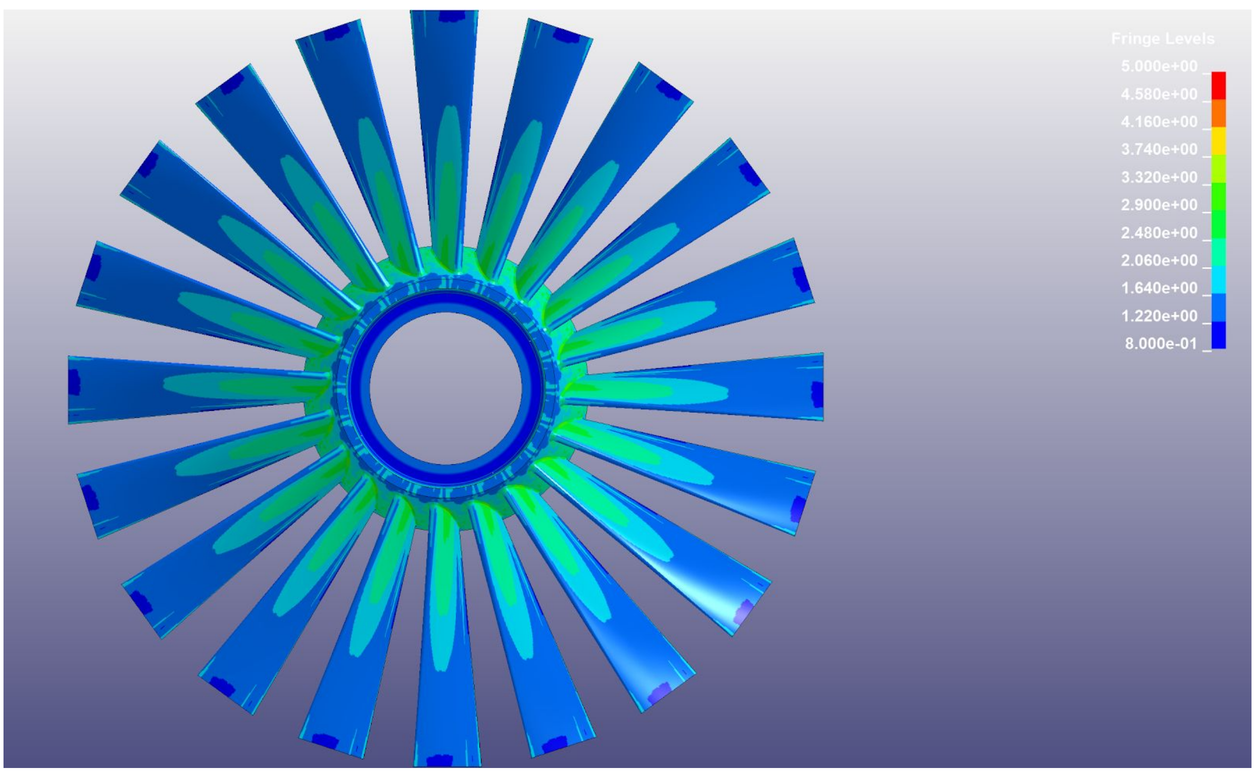

Figure 9: Aspect Ratio in New Mesh

The completed blisk assemblies for the old and new meshes are given in Figures 10 and 11.

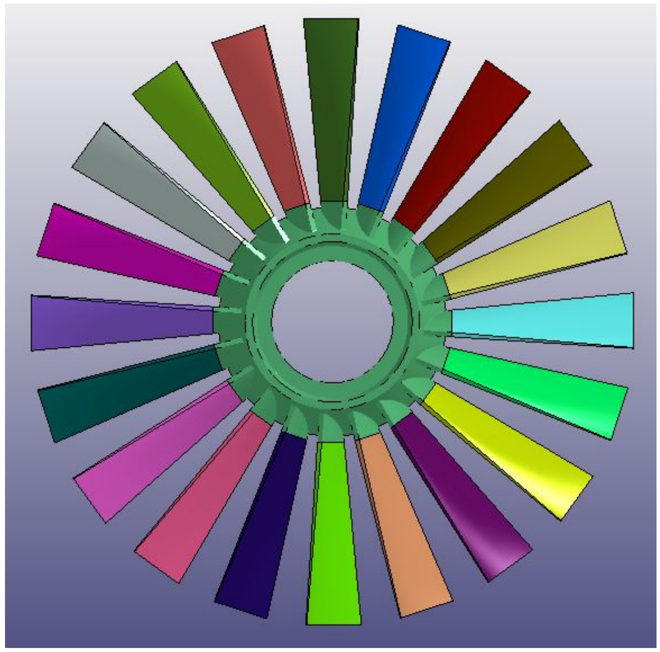

Figure 10: Old Blisk Assembly

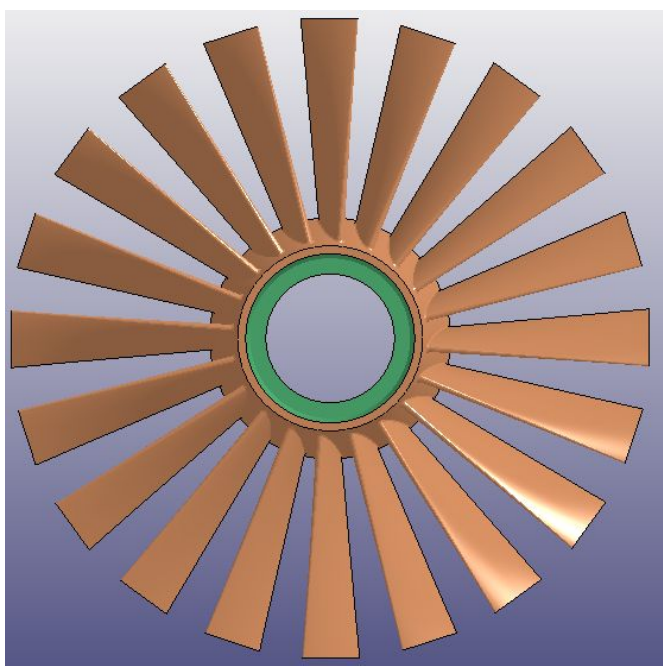

Figure 11: New Blisk Assembly

In these assemblies, each different color represents a different part, so the new mesh contains only two parts while the old mesh contains 21 parts. The mesh for the inner disk from the new blisk (visible in green in Figure 11) is given in Figure 12.

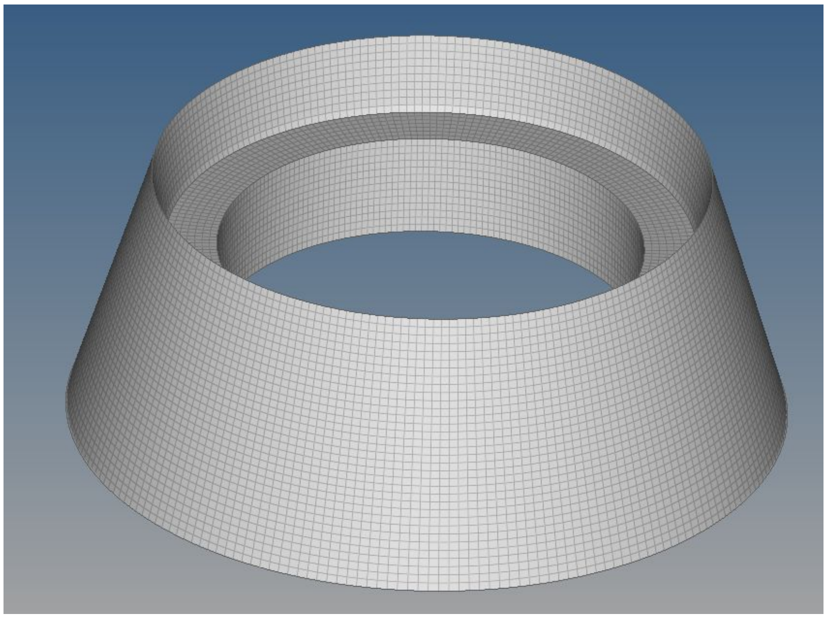

Figure 12: Inner Disk Mesh

As a result of the new mesh, the number of elements needed has changed for both the thin and thick geometries. The elements counts for all combinations are given in Table 1.

Table 1: Blisk Mesh Element Counts

Geometry	New Mesh	Old Mesh
Thick	1,294,100	1,290,772
Thin	2,107,460	1,728,181

Table 1 shows that the thick mesh had almost no change in element count while the thin mesh increased by about 400,000 elements. This is due to the increased number of elements in the thickness for the thin mesh and improving the aspect ratio of elements throughout the model. Achieving 4 elements in the thickness for the thin mesh required the use of elements smaller than

1 mm, thus creating a larger increase in element count than the thick blade when compared to their respective old meshes.

A new mesh for the nose cone (or spinner) was also created. Like the blisk mesh, this mesh contained primarily hexahedron elements with a small number of pentahedrons as needed. The previously used mesh for the nose cone was constructed entirely out of tetrahedron elements, so the switch to hexahedron elements should significantly improve the quality of impact results obtained from the nose cone. It is determined that this mesh requires 4 elements in the thickness to stay below the 1 mm size target set out for this project's meshes.

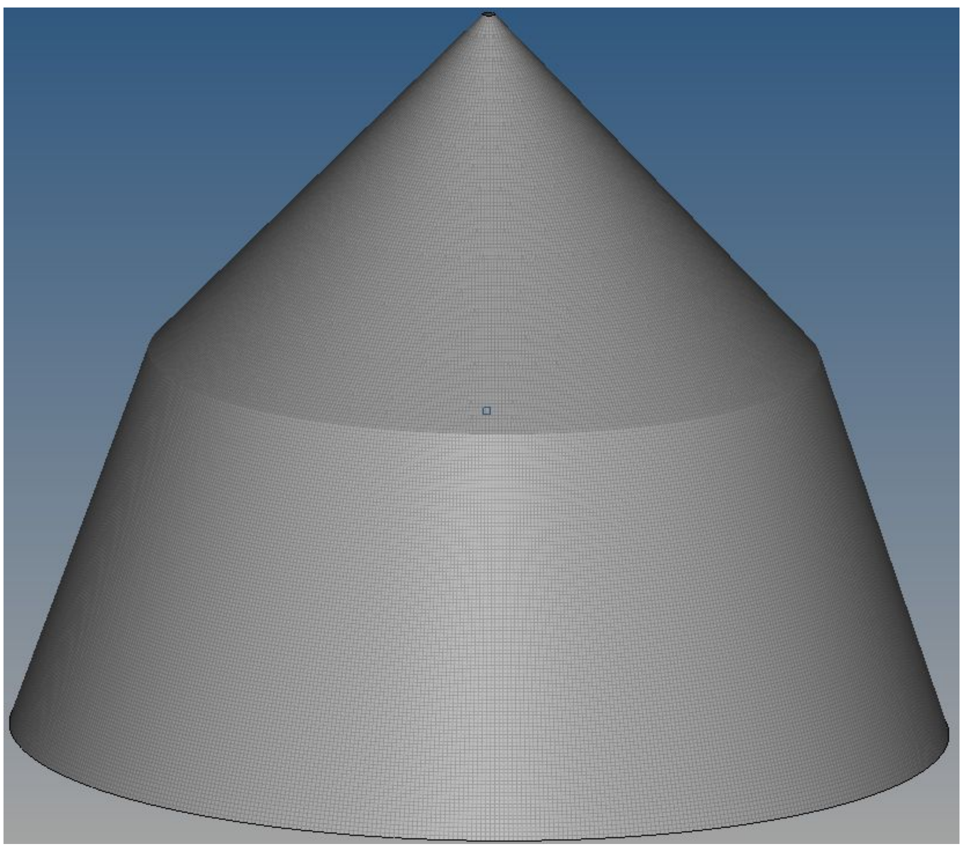

Figure 13: Nose Cone Mesh

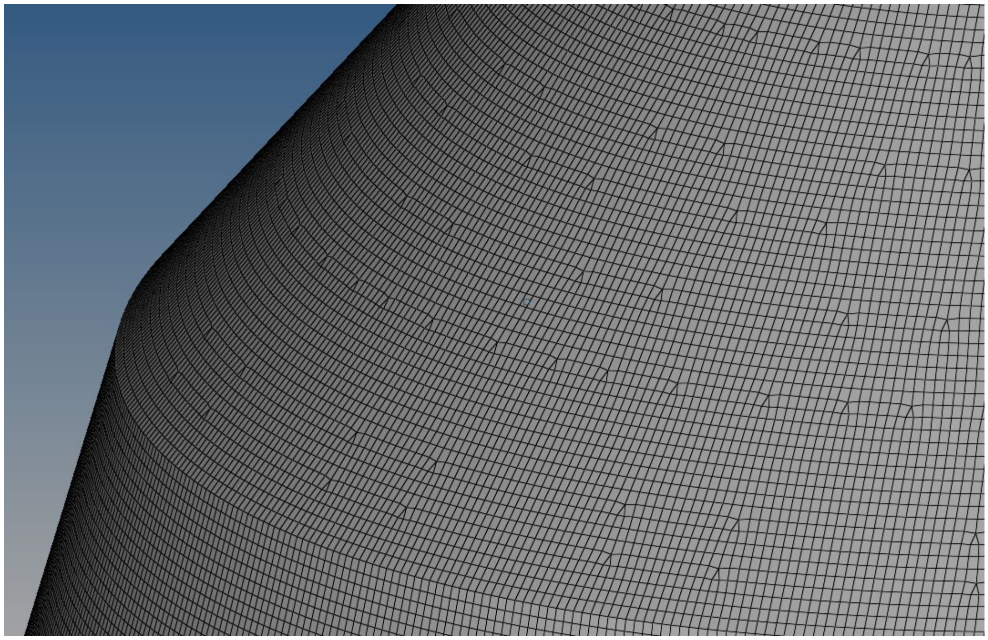

Figure 14: Nose Cone Mesh Closeup

From these figures, the predominantly hexahedron construction of the mesh is visible. The mesh of the nose cone contains 443,920 elements.

3.1.2 Mesh Quality Testing

With the problems from the old mesh fixed, the next challenge is to get a new blisk mesh that does reasonably well in mesh quality checks. Unlike the nose cone mesh, which had no problem meeting the specified quality parameters, the blisk mesh proves challenging to create in a way that keeps mesh quality high. As mentioned in §2.1, the quality parameters being checked are Jacobian, aspect ratio, and warp angle. Ultimately, these quality checks were able to be met everywhere except in some elements comprising the blade base fillet. The fillet was included in the model to mitigate any stress concentration that might occur there, and given that most elements in the base were within quality guidelines and the base won't experience direct impacts, it was decided that this was an acceptable trade off.

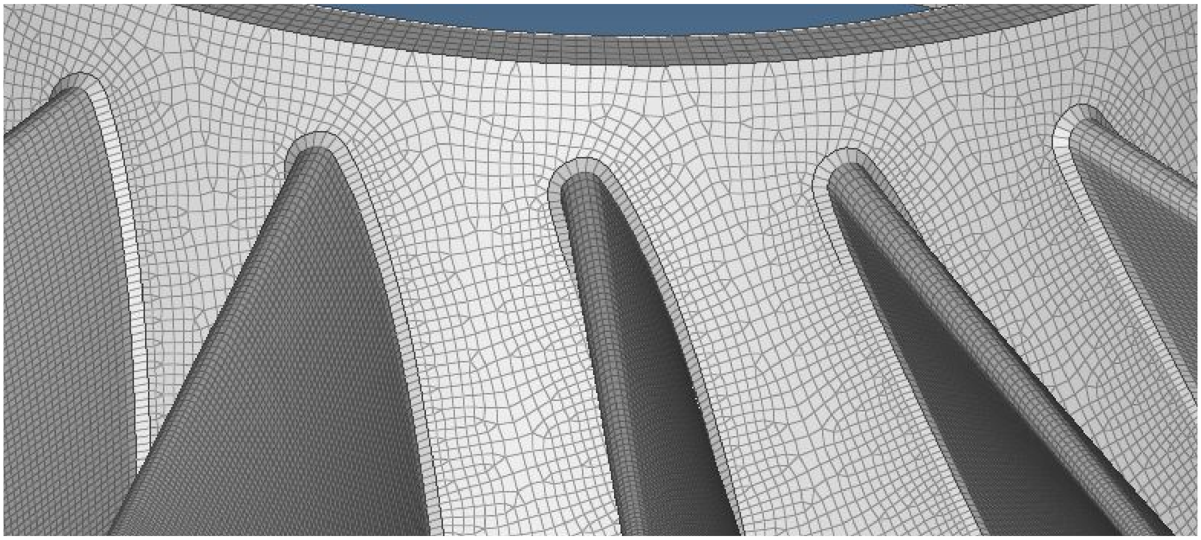

Figure 15: Elements in the Blade Base Fillet

Table 2 shows the number of elements per blade that fail quality checks in each of the new meshes. The desired values are specified as >0.5 for Jacobian, <5 for aspect ratio, and <30° for warp angle.

Table 2: Failed Quality Check Elements per Blade

Blade Geometry	**Jacobian**	**Aspect Ratio**	**Warp Angle**
Thick	5	0	27
Thin	11	11	42

Figure 16 shows where the failing elements are located for the thin blade warp angle, the worst case encountered. The figure shows that the number of elements with quality issues is relatively small, even when looking at just the base layer of the blade. The failing elements are highlighted in white.

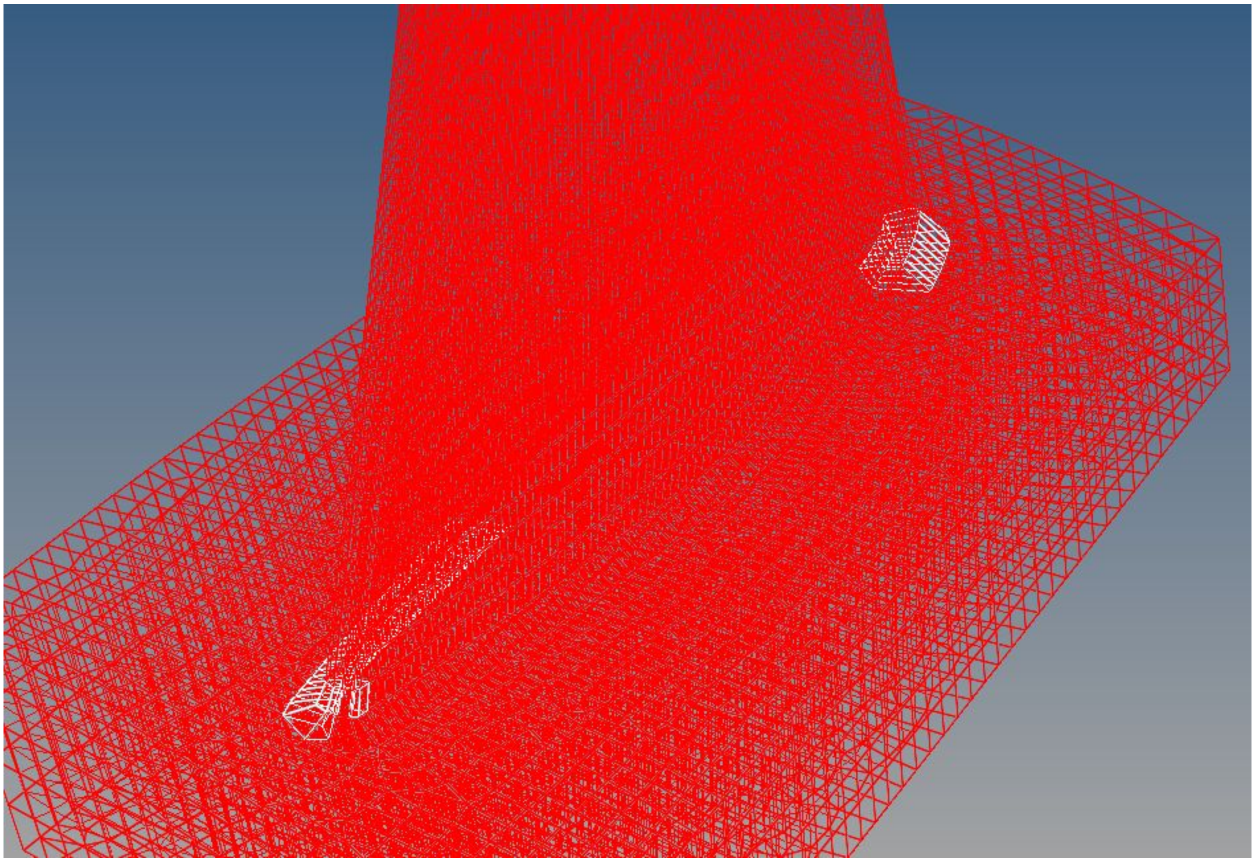

Figure 16: Elements Failing Warp Angle Quality in the Thin Blade

In conclusion of the mesh creation phase of the project, the new meshes created succeed in remedying multiple issues previously identified with the old mesh of the fan geometry. The new meshes also in many cases improve the quality of the elements while moving to a hexahedron mesh.

3.2 Comparison of Impact Results

To better gauge the improvements achieved with the new mesh, impact tests are conducted on the thick fan mesh in LS-DYNA. As mentioned in §2.2, the impact object used for this testing is a hardened steel bolt. The goal of these tests are to observe the expected stress discontinuity in the old mesh and how the new mesh fixes this, as well stress and plastic strain in the area of the bolt impact.

3.2.1 2000 RPM Results

For the 2000 RPM tests with the old and new meshes, both bolts make glancing blows causing only minor damage to the blade. Some elements in both meshes fail, but no severe structural damage is seen in either case.

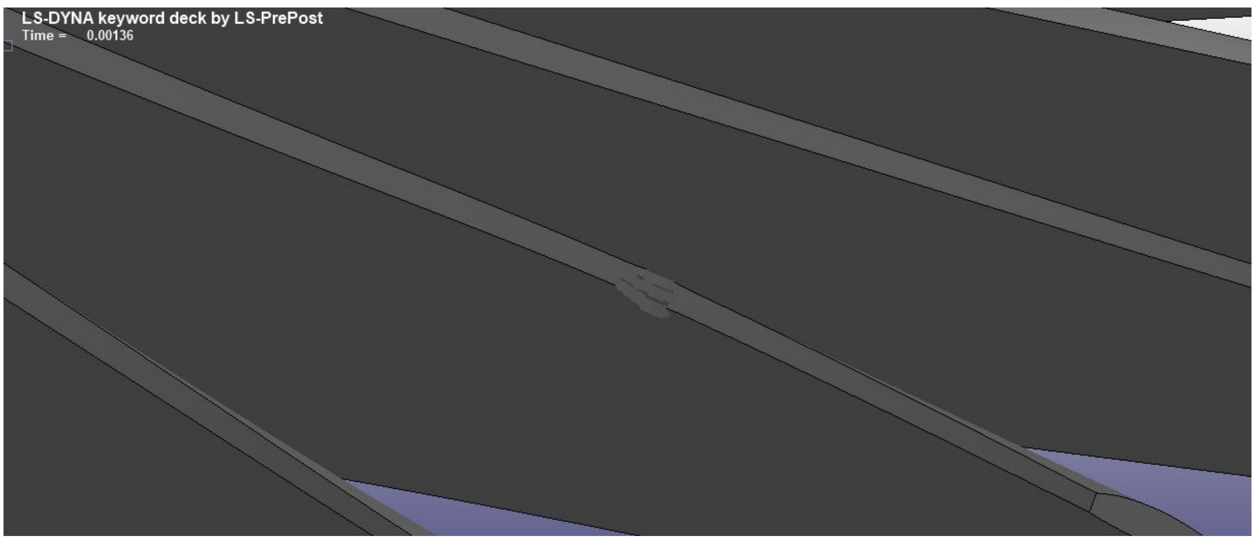

Figure 17: Old Mesh 2000 RPM, Damage to Leading Edge

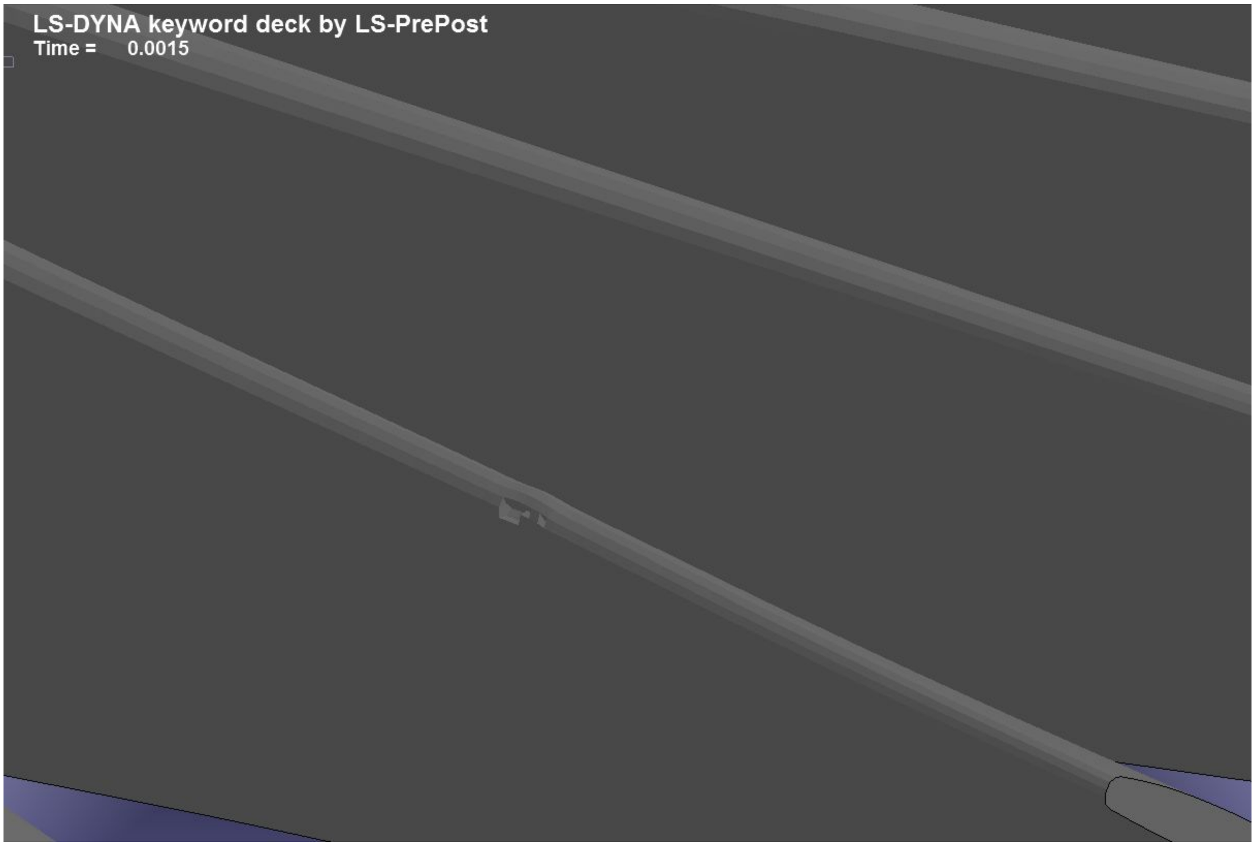

Figure 18: New Mesh 2000 RPM, Damage to Leading Edge

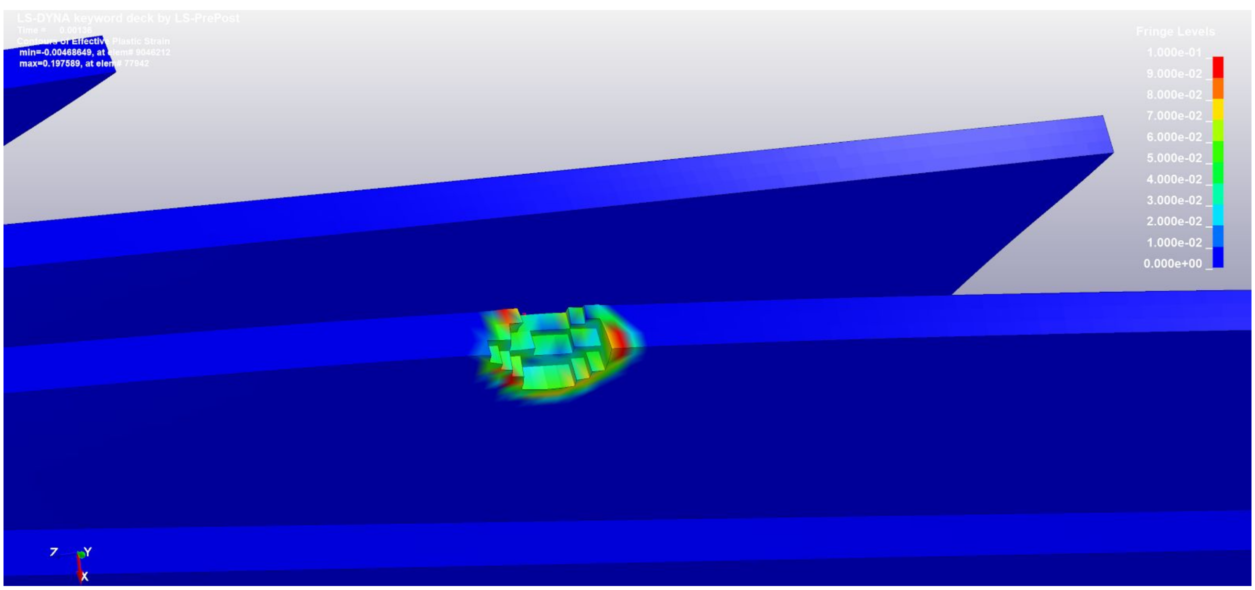

Figure 19: Old Mesh 2000 RPM, Effective Plastic Strain

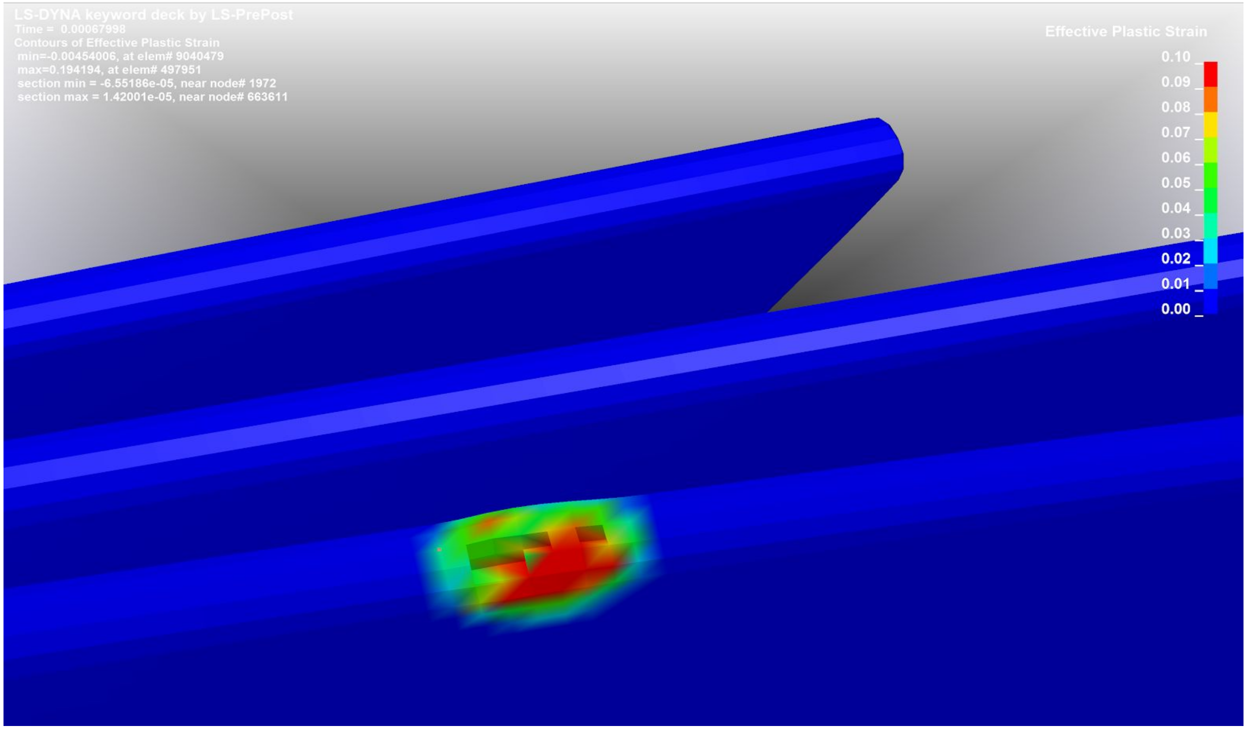

Figure 20: New Mesh 2000 RPM, Effective Plastic Strain

The damage in these cases is not identical due to subtle differences in the bolt's position when impact occurred, but the general class of damage is similar. The strain contours in these images also show that the elements adjacent to those that failed are either right at or just below an effective plastic strain of 0.1. Because the MAT-224 model calculates failure based on accumulated effective plastic strain [7], this indicated that both meshes are modelling failure accurately.

The Von Mises stress contours during both impacts are also checked as a way of comparing the response of both meshes to impact loading.

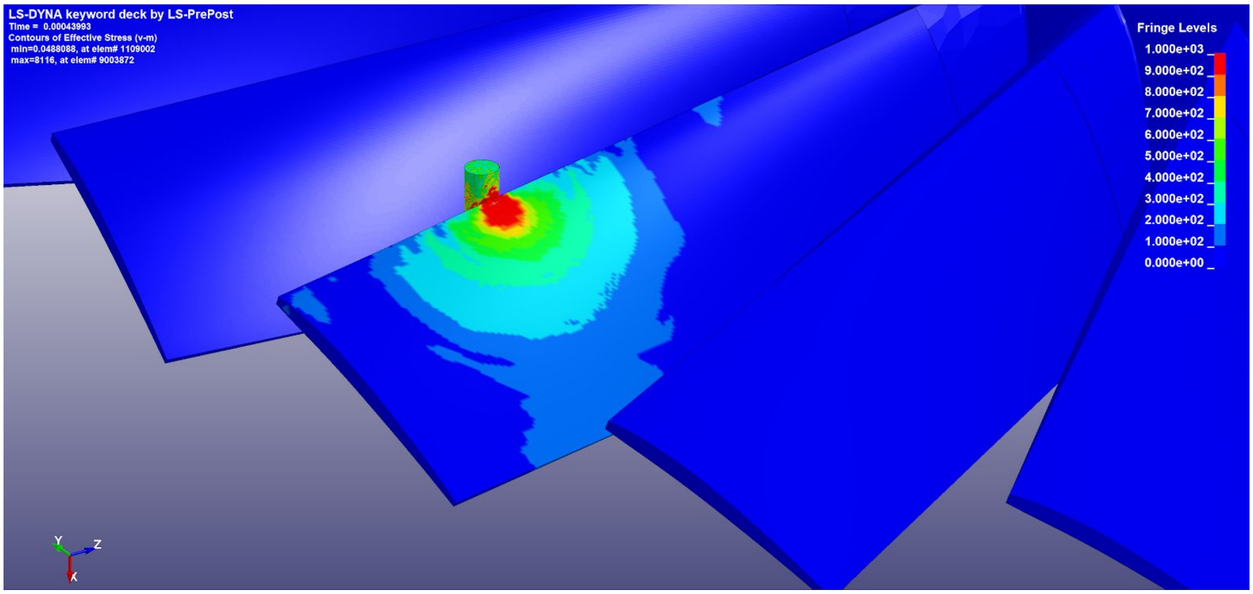

Figure 21: Old Mesh 2000 RPM, V-m Stress During Impact

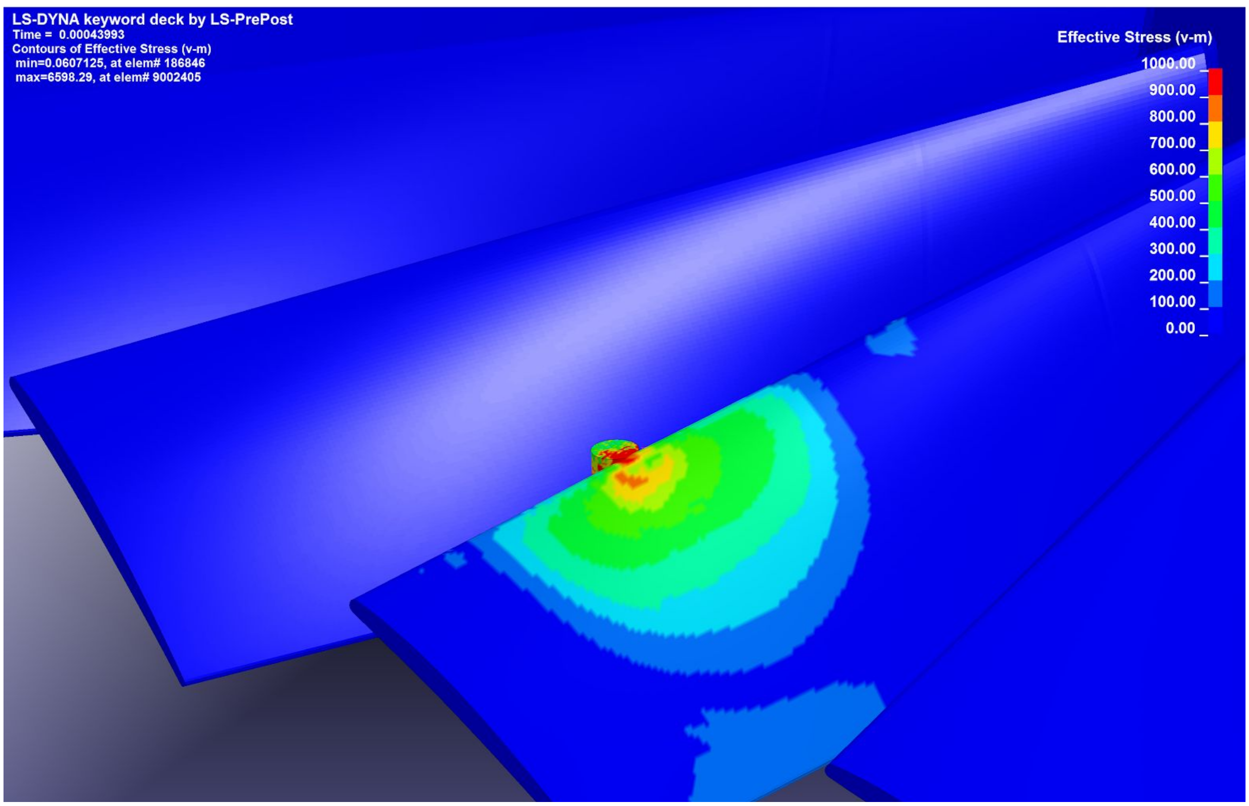

Figure 22: New Mesh 2000 RPM, V-m Stress During Impact

As with the figures of plastic strain, the Von Mises stress shows that the two meshes behave broadly the same under impact loading. One improvement that the new mesh makes is visible in these V-m plots. Unlike the old mesh, which has a square leading edge, the new mesh has a rounded leading edge. The smoother V-m contours along the new mesh's leading edge come as a result of this change. This change helps better resolve contact behavior in LS-DYNA, which should assure more consistency when running simulations.

With the blades' impact loading behaviors compared, the effect of the old mesh's contact in the blade is now examined. To do this, the stress contours from both impacts are examined in the region of the old mesh's contact after enough time has passed for the impact wave to propagate to the blade root.

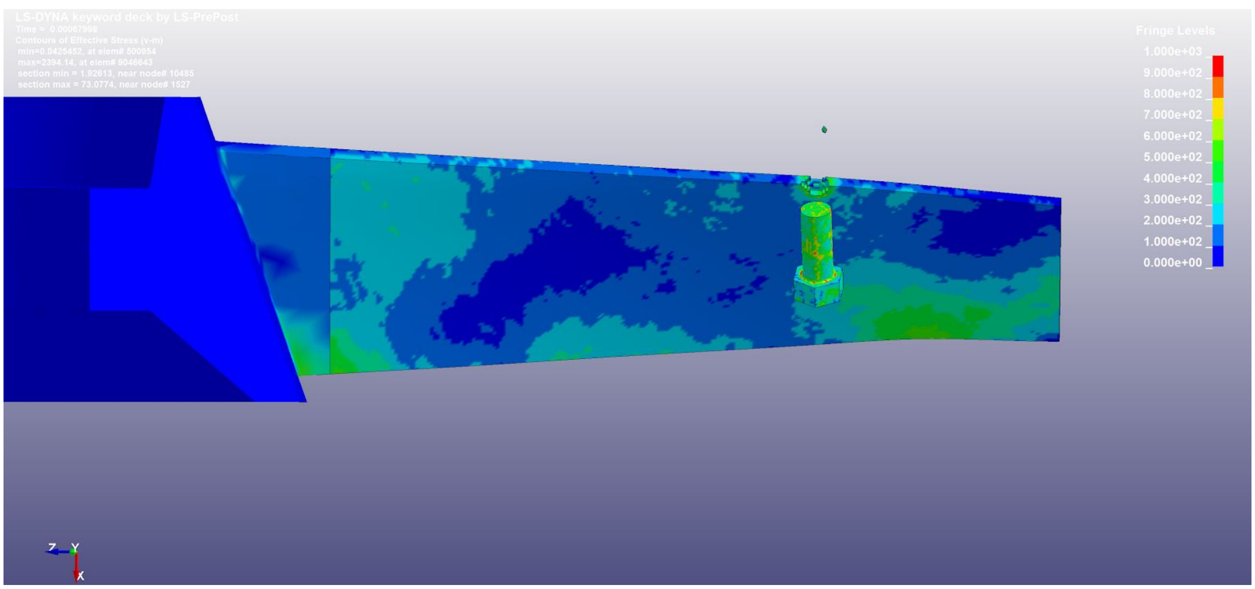

Figure 23: Old Mesh 2000 RPM, V-m Stress After wave Propagation

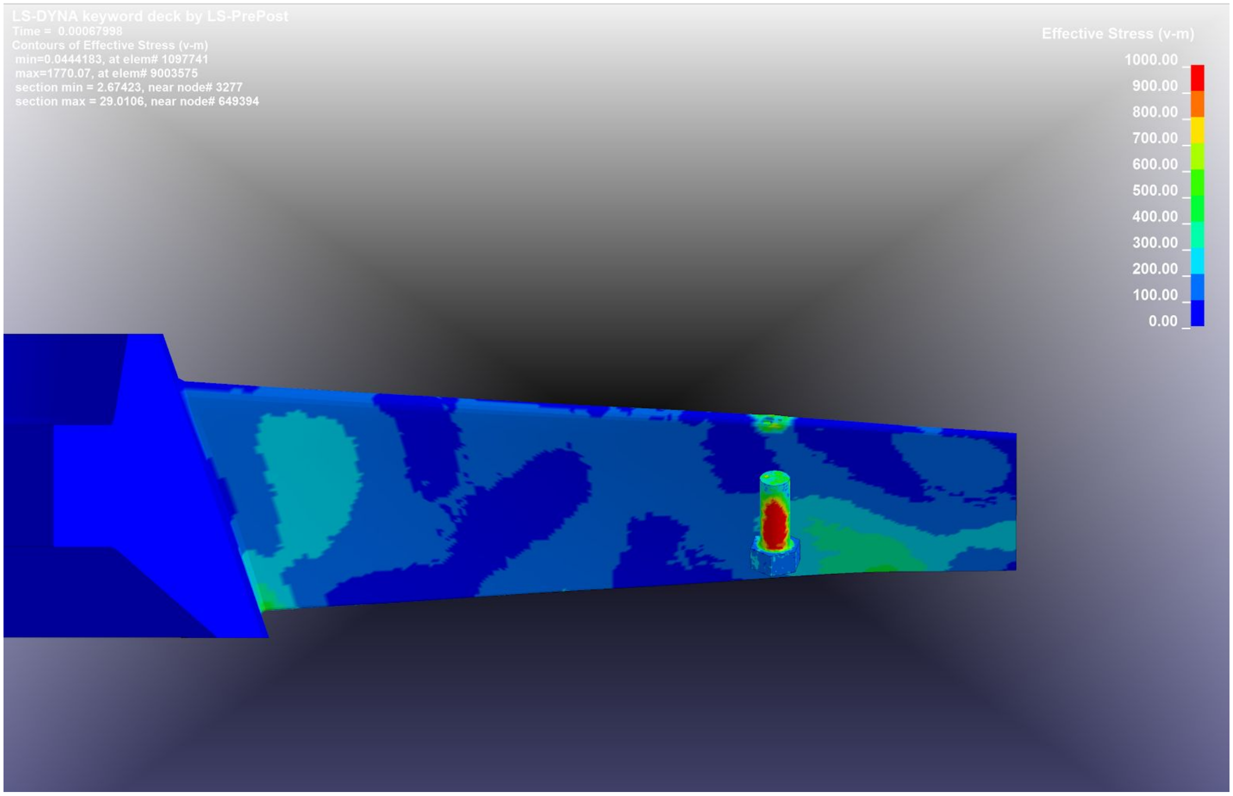

Figure 24: New Mesh 2000 RPM, V-m Stress After Wave Propagation

The contours in the old mesh clearly show the presence of a stress discontinuity at the location of the blade contact. While this doesn't directly affect the quality of the impact results tested here, it does reduce the accuracy of how stress is transferred to the fan disk, and it may negatively affect impact results for impacts that occur further in on the blade. As expected, the new mesh has no stress discontinuity over the length of the blade, and stress can be observed transferring smoothly to the fan disk.

3.2.2 8500 RPM Results

Compared with the 2000 RPM results, the tests for 8500 RPM yield far greater damage to the blades. Tests with both the new and old meshes resulted in partial loss of one blade with significant damage being inflicted on another.

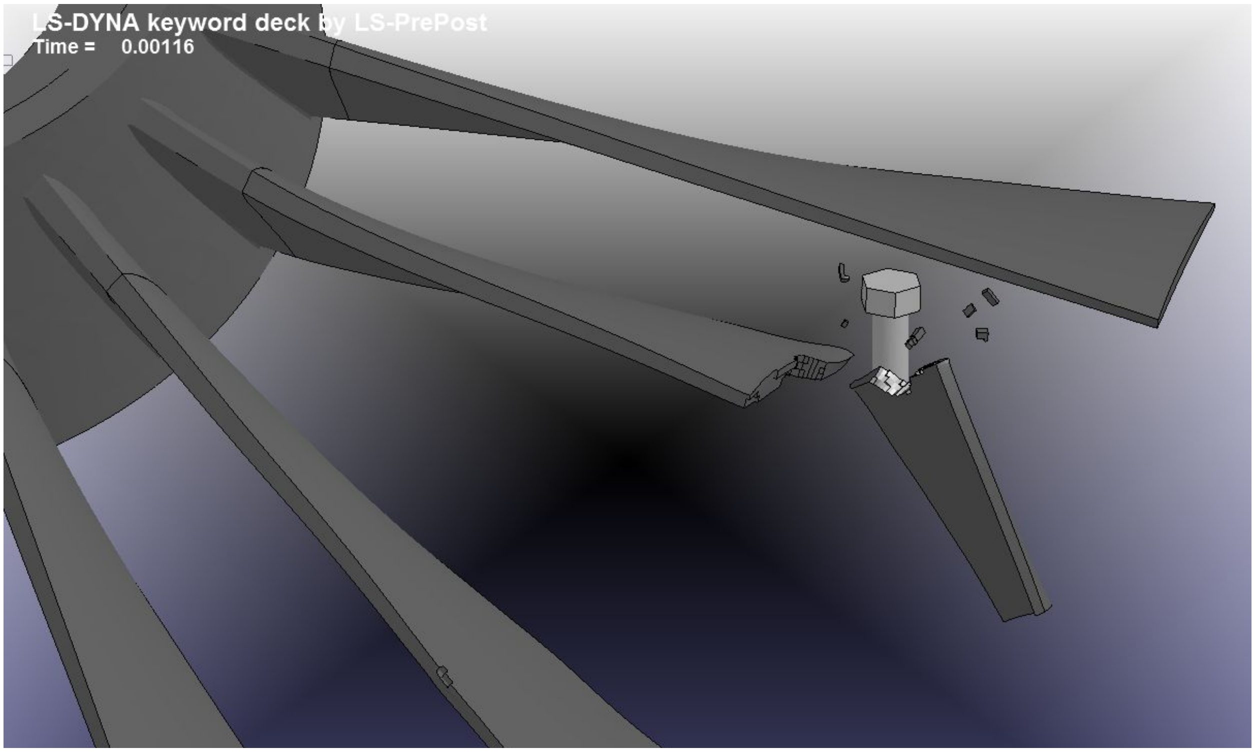

Figure 25: Old Mesh 8500 RPM, Partial Blade Out

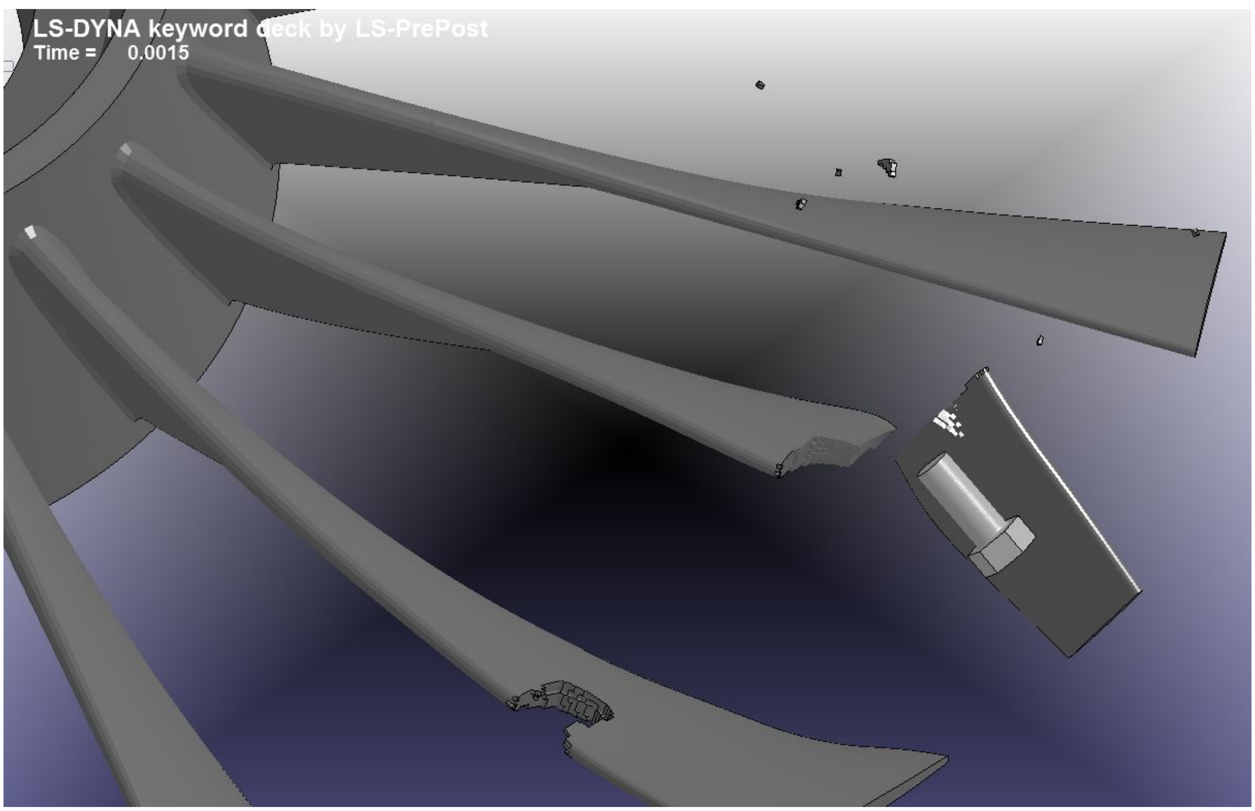

Figure 26: New Mesh 8500 RPM, Partial Blade Out

Due to slight variation in the bolt's position for the two simulations, the bolt skipped off of the first blade it encountered in the test of the old mesh, and the bolt caused significant damage to the first blade in the test of the new mesh. As a result, the bolt impacted the second blade with a higher relative velocity in the test of the old mesh and continued on to impact a third blade while the bolt came to a stop in the test of the new mesh after impacting the second blade. In both cases, the second blade was cut approximately halfway through its chord by the bolt and then the second half ripped off. Overall, the damage caused by both bolts was very similar. This can be seen in the Figures 27 and 28, which display the elements that failed in each model.

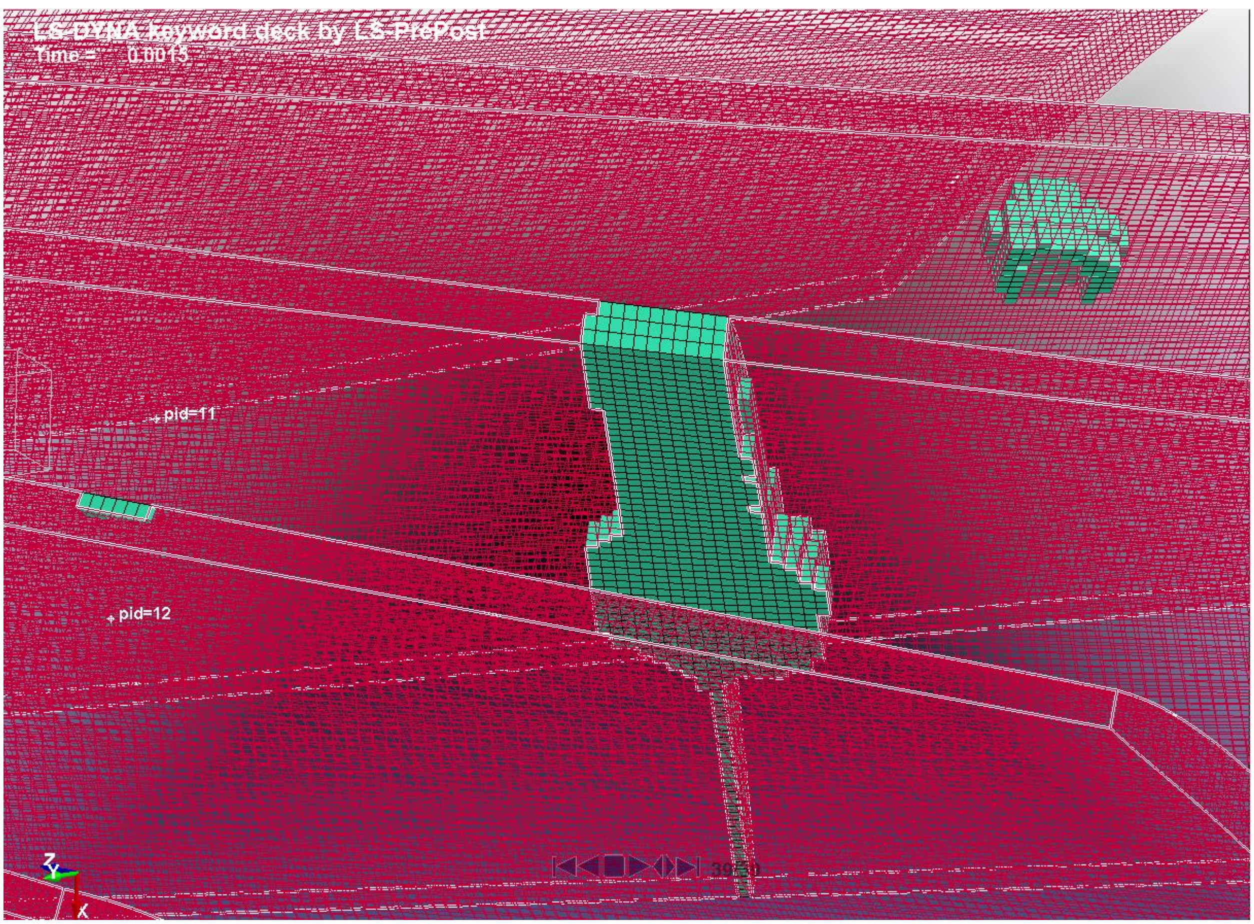

Figure 27: Old Mesh 8500 RPM, Failed Elements

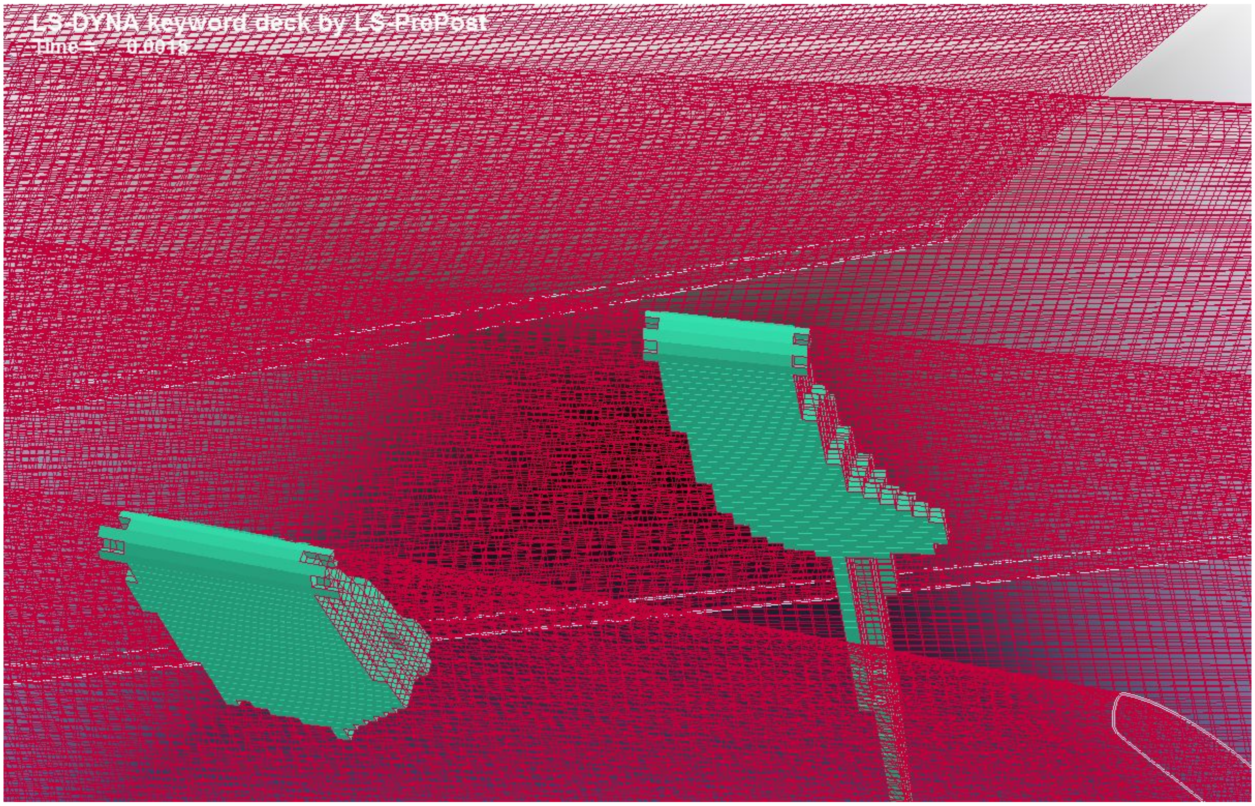

Figure 28: New Mesh 8500 RPM, Failed Elements

These figure show that although the bolts inflict damage in different places, the overall amount of damage caused is similar for both. Note the clear distinction in number of elements which failed along the chordwise direction of the second blade. This is caused by the change of failure modes as the bolt's cutting effect becomes dominated by the crack propagation of the blade under its own inertia.

Once again a discontinuity in Von Mises stress was encountered across the contact in the old mesh. This further validates the improvements that the new mesh has made when compared with the old mesh.

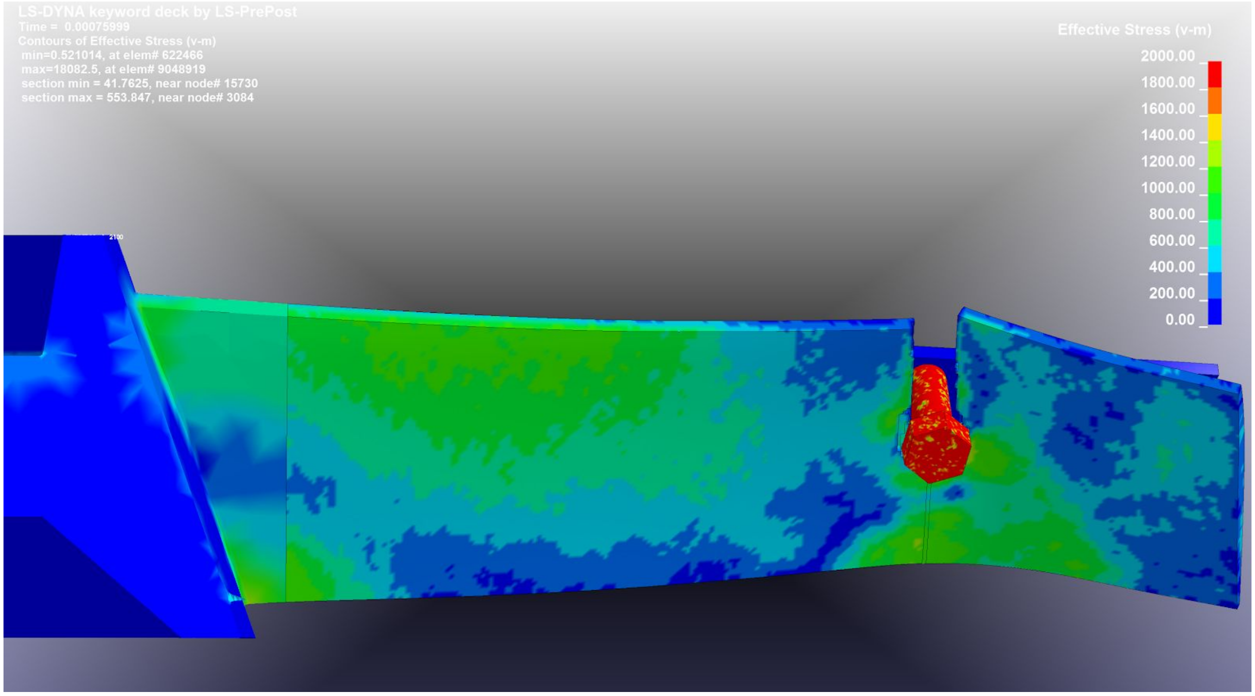

Figure 29: Old Mesh 8500 RPM, Stress Discontinuity

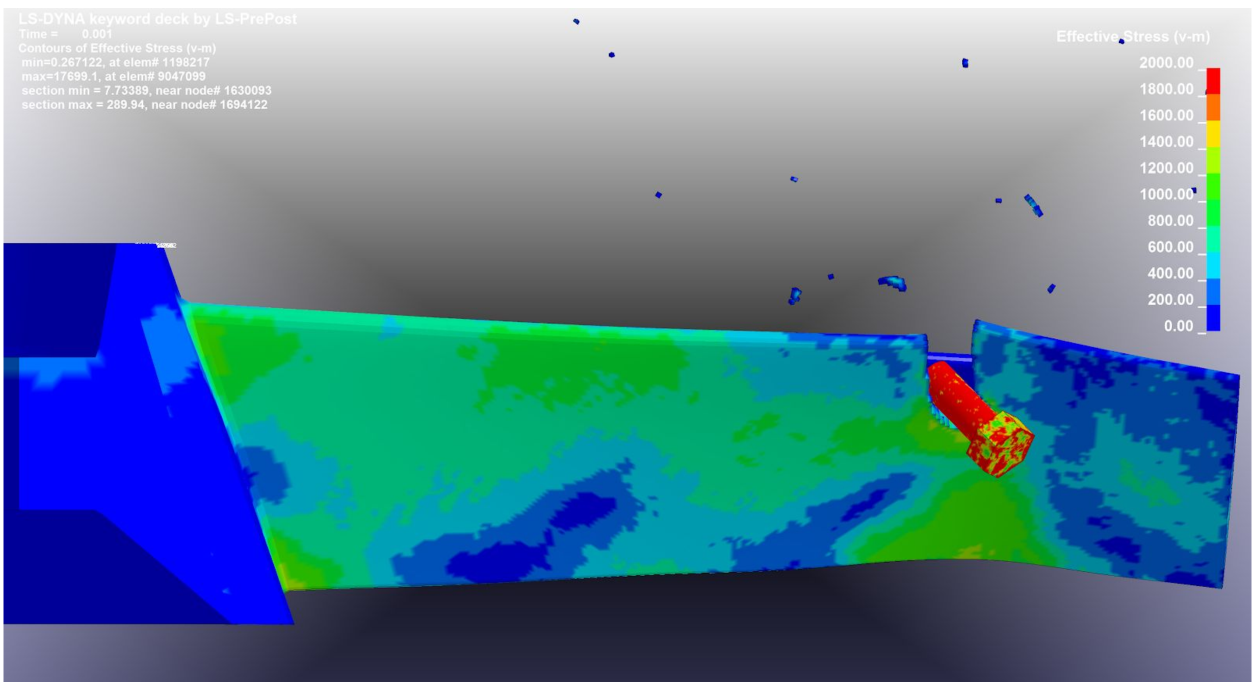

Figure 30: New Mesh 8500 RPM, Stress Discontinuity

Note that there is a stress discontinuity across the contact within the disk of the new mesh. However, the stress here is far lower than in the blades where the old mesh's contact is located, and, as can be seen in Figures 31 and 32, the new mesh is able to eliminate the strain discontinuity that is visible in the old mesh.

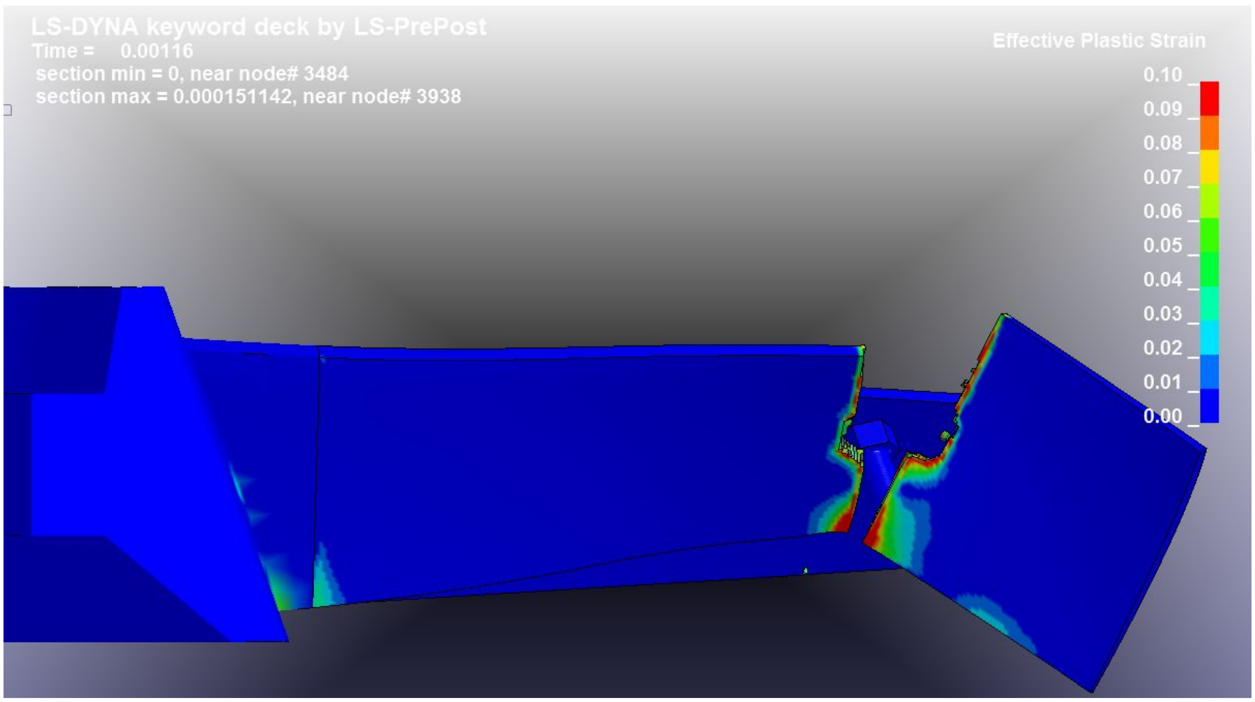

Figure 31: Old Mesh 8500 RPM, Strain Discontinuity

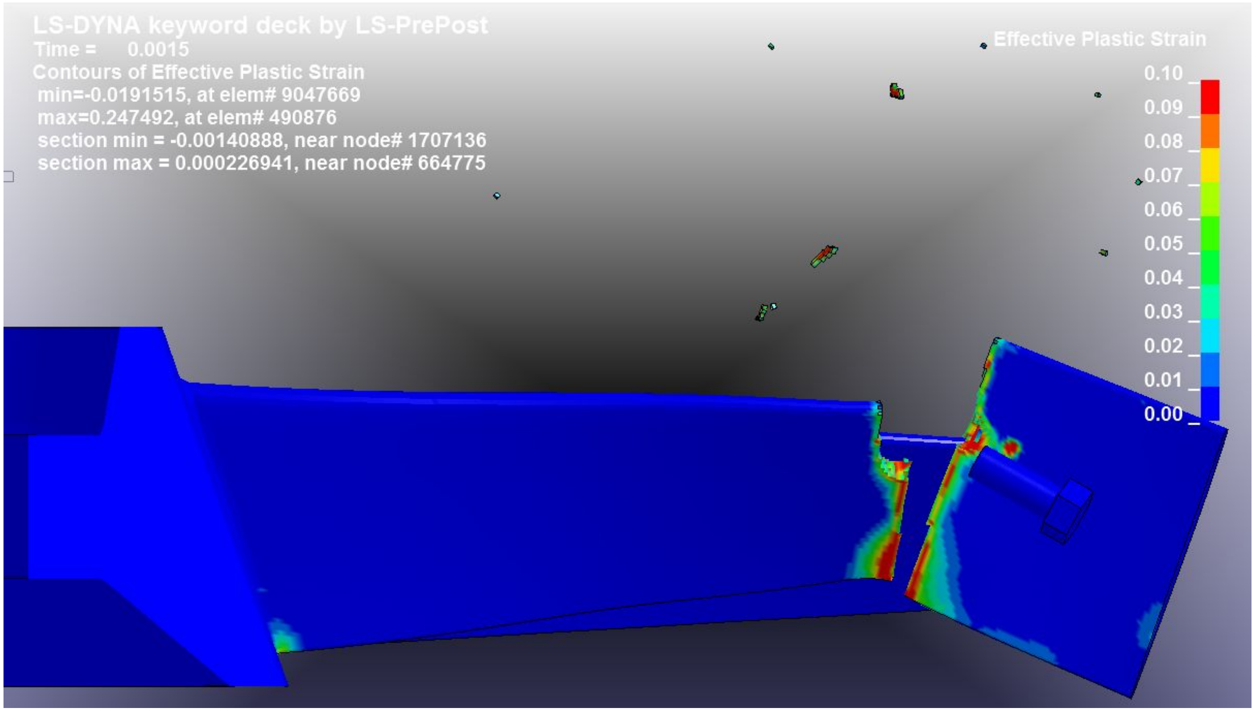

Figure 32: New Mesh 8500 RPM, Strain Discontinuity

These figures show that there exists a concentration of effective plastic strain along the the lower edge of the blade contact in the old mesh. This concentration could potentially cause improper failure of the blade under extreme loading conditions and contacts in the blades should be avoided for this reason.

As with the 2000 RPM tests, the effective plastic strain is also examined in the impact region to verify that failure in both the new and old meshes is occurring consistently with one another.

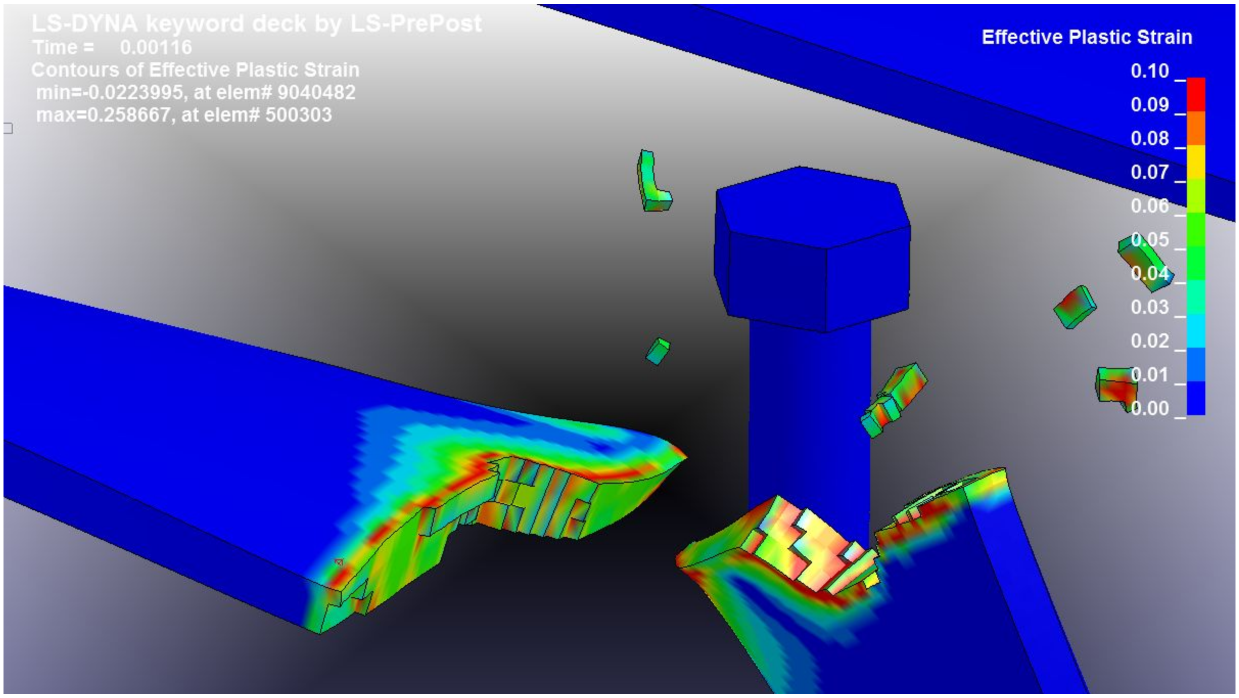

Figure 33: Old Mesh 8500 RPM, Effective Plastic Strain Along Fracture Line

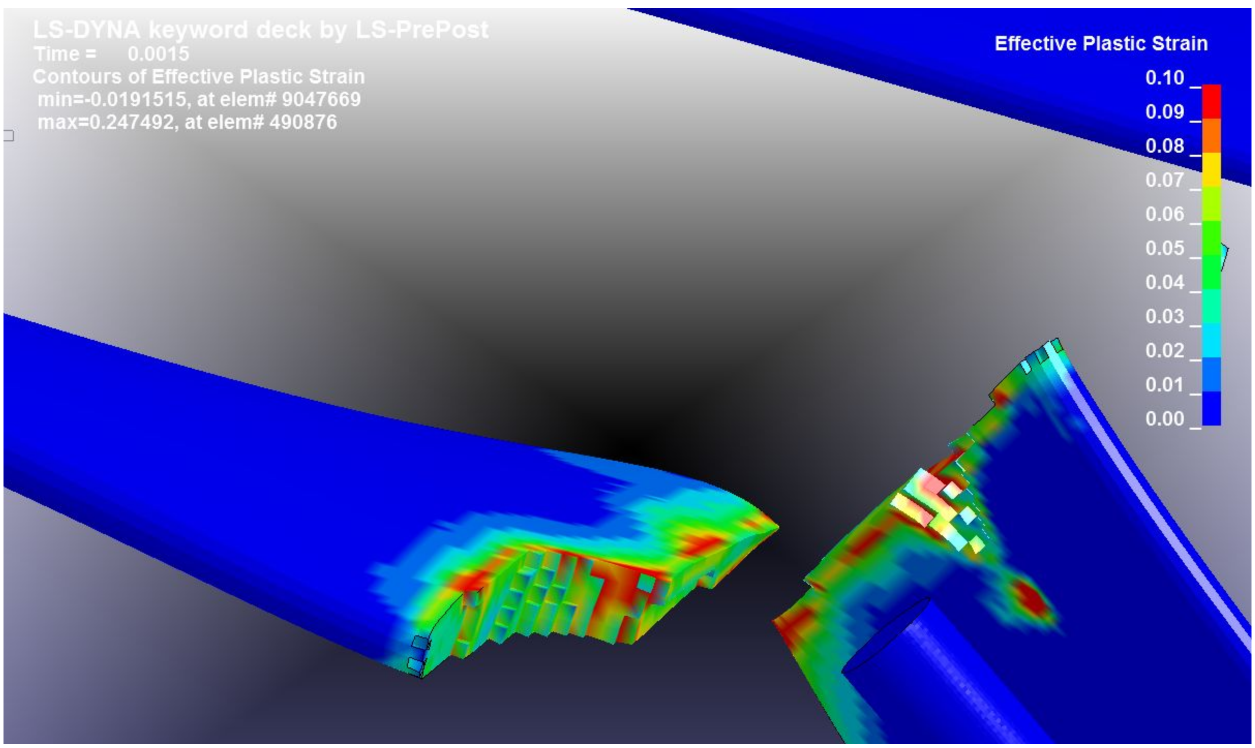

Figure 34: New Mesh 8500 RPM, Effective Plastic Strain Along Fracture Line

As with the 2000 RPM tests, the strain contours of both simulations agree closely with one another. This indicates that the MAT-224 failure model is working as it should.

4. Summary and Conclusions

In this project, new meshes are created for use in future research into the damage caused to a fan by UAV ingestion. The primary goals of creating hexahedron meshes and removing contacts from the blades are successfully completed. To better gauge the improvements made by the new mesh, impact tests are conducted using a steel bolt at the impactor. These impact tests show that there was a stress discontinuity present in all impacts using the old mesh, and there was a discontinuity in effective plastic strain in the high energy 8500 RPM test conducted. The new meshes solve all of these problems, meets quality standards, and shows similar damage to the previous fan mesh under hard body impact. These improvements will allow for greater fidelity in future simulations, aiding in the continued work of the GTL to produce highly accurate simulations of UAV ingestion using LS-DYNA.

5. Future Work

A primary focus moving forward will the the extension of the MAT-224 Ti-6Al-4V mesh regularization curve. As mentioned in §1.2, the MAT-224 model requires a mesh regularization curve that is inclusive of all mesh sizes in the mesh being used to achieve its full potential. The current mesh regularization curve for the titanium model used in the fan includes elements from 0.1 to 0.4 mm [8]. Elements of this size are far too small to be used in the meshing a full sized fan while maintaining a reasonable number of elements, and attempts to create meshes of the fan using this sizing resulted in meshes with more than 30 million elements. This also drastically decreases the mass of the smallest element, forcing LS-DYNA to use much smaller time steps and increasing computational cost. This leaves extending the mesh regularization for the titanium model as the only feasible option.

Extending the model is achieved by matching the failure of dogbone specimens under tension in LS-DYNA to experimental data using the same dogbone [7]. Initial work has taken place in the creation of these dogbone specimens to be used in extending the mesh regularization curve.

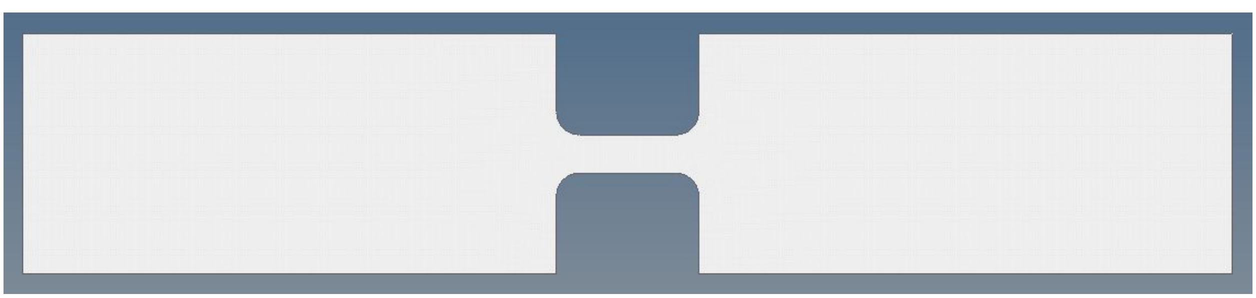

Figure 35: Dogbone Specimen

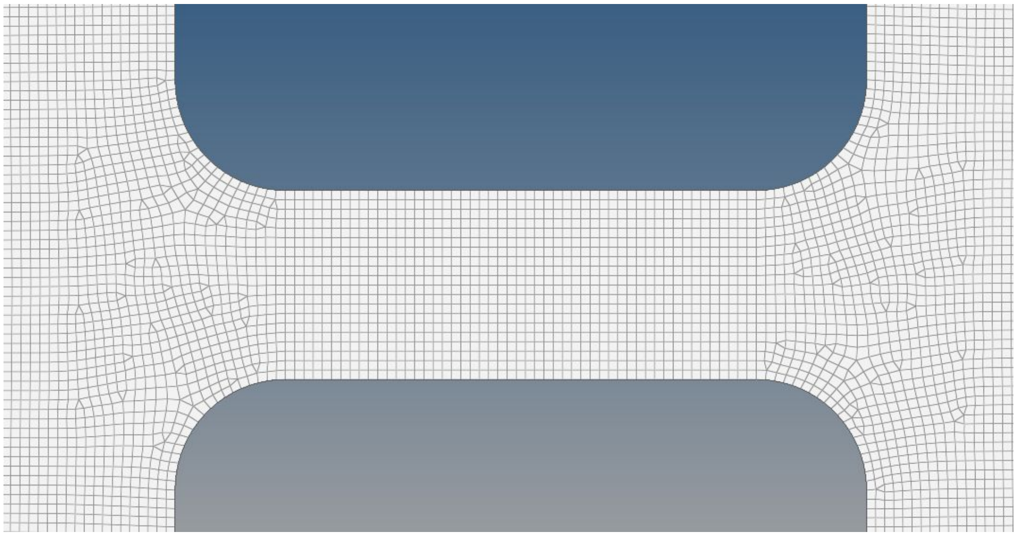

Figure 36: Close Up of Mesh for Dogbone, 0.1 mm

The dogbone being used is matched to dimensions of the dogbone from the experimental data with which it is being compared [9]. Meshes are then created using 0.1, 0.2, and 0.4 mm elements, the element sizes used to create the mesh regularization curve originally [8]. These meshes of the experimentally matched dogbone will be used to verify that the MAT-224 model is working properly.

With the model verified, a larger dogbone must be created to allow enough space for the larger elements being validated (only two elements were used in the thickness for the 0.4 mm elements [8], making this specimen far too small for testing elements of the desired size). This larger dogbone will then be tested with the 0.1, 0.2, and 0.4 mm elements to confirm that their failure points still agree. With this confirmed, elements of 1, 2, 3, and 4 mm will be tested and matched to the regularized failure points of the 0.1, 0.2, and 0.4 mm elements. This will allow for future simulations using the MAT-224 Ti-6Al-4V model to use elements anywhere from 0.1 mm to 4 mm while still maintaining the highest degree of accuracy.

References

[1] United States, Congress, Federal Aviation Administration, and Peter A White. "Faa.gov."*Faa.gov*, 7 Aug. 2009. www.faa.gov/documentLibrary/media/Advisory_Circular/AC_33_76-1A.pdf.

[2] Plenumchamber. "JT8D Engine after Bird Strike." *Wikipedia*, Wikimedia Foundation, Inc., 11 July 2007, commons.wikimedia.org/wiki/File:JT8D_Engine_after_Bird_Strike.jpg.

[3] United States, Congress, Federal Aviation Administration, et al. "Volume IV – UAS Airborne Collision Severity Evaluation – Engine Ingestion." *Volume IV – UAS Airborne Collision Severity Evaluation – Engine Ingestion*, Federal Department of Transportation, Aug. 2017. www.assureuas.org/projects/deliverables/a3/Volume%20IV%20-%20UAS%20Airborne%20Collision%20Severity%20Evaluation%20-%20Engine%20Ingestion.pdf.

[4] Wang, Erke, et al. "Back to Elements - Tetrahedra vs. Hexahedra ." CAD-FEM GmbH.

[5] "Modeling Guidelines Document." LS-DYNA Aerospace Working Group, 23 Sept. 2017.

[6] "Element Quality and Checks." Altair.

[7] Buyuk, Murat. "Development of A Tabulated Thermo-Viscoplastic Material Model with Regularized Failure for Dynamic Ductile Failure Prediction of Structures under Impact Loading." *PQDT Open*, *The George Washington University*, 2013, pqdtopen.proquest.com/doc/1458428327.html?FMT=ABS.

[8] United States, Congress, Federal Aviation Administration. "Development of a Titanium Alloy Ti-6Al-4V Material Model Used in LS-DYNA ." *Development of a Titanium Alloy Ti-6Al-4V Material Model Used in LS-DYNA* , Federal Department of Transportation, May 2016. www.tc.faa.gov/its/worldpac/techrpt/tc15-23.pdf.

[9] Hammer, Jeremiah. "Plastic Deformation and Ductile Fracture of Ti-6Al-4V under Various Loading Conditions." *OhioLink*, *The Ohio State University*, 2012, etd.ohiolink.edu/!etd.send_file?accession=osu1354700435&disposition=inline.